The Design and Testing of a Radial Flow Turbine for Aerodynamic Research

I. HUNSTMAN & H. P. HODSON
Whittle Laboratory
Cambridge University
England

S. H. HILL
Rolls-Royce plc
Leavesden
England

ABSTRACT

This paper describes the design of a high-speed radial inflow turbine for use as part of a gas-generator, and the design of a large-scale (1.2 m tip dia.) low-speed model of the high-speed turbine. Stream-line curvature through-flow, two-dimensional blade-to-blade and fully three-dimensional inviscid and viscous calculation methods have been used extensively in the analysis of the designs. The use of appropriate scaling parameters and their impact on turbine performance is discussed. A simple model shows, for example, how to model the blade lean in the inducer which serves to balance the effect of meridional curvature at inlet to the rotor and can be used to unload the rotor tip. A brief description of the low speed experimental facility is followed by a presentation and discussion of experimental results. These include surface flow visualisation patterns on both the rotor and stator blades and blade row exit traverses.

INTRODUCTION

Radial inflow turbines offer several advantages for use in small turboshaft applications when compared with axial turbines for the same duty. This is because the radial inflow turbine offers greater work extraction per stage at comparable or higher efficiencies, increased ruggedness, lower costs of manufacture and improved packaging when used in conjunction with a reversed flow combustor.

Studies at Rolls-Royce have shown that a cooled, high-efficiency radial turbine could offer significant improvements in performance as the gas-generator turbine of a high technology turboshaft engine, if small improvements in current levels of technology could be achieved. An un-cooled radial turbine of a similar aerodynamic duty would also present an attractive proposition in smaller power-plants. However, the problems facing the designers of today's radial turbines are not inconsiderable, particularly in the areas of rotor cooling and rotor aerodynamics.

Many of the published design methods are largely based on the design rules developed by NASA and others (e.g. Hiett and Johnston (1963), Rohlik (1975) and Glassman (1976)). These methods or their adaptations for specific applications are essentially based on observations of the overall performance of radial turbines. Very few are based on observed physical processes even though many of the flow and loss models purport to model the details of the flow. In such circumstances, the possession of one-dimensional information and models places potentially unnecessary restrictions on the design process. Today, commercial organisations are replacing costly experimental development programmes by the use of modern CFD codes such as the viscous analysis code of Dawes (1986) but until the reasons behind such phenomena as 'incidence shock-loss' or tip clearance losses are understood and error will play a large part in any design process.

This paper describes the initial stages of a research and development programme in radial turbines which addresses the identification and understanding of the major sources of loss and the assessment of aerodynamic design and analysis methods. A large-scale, low-speed radial inflow model turbine has been constructed at the Whittle Laboratory, Cambridge University as part of the research programme. The model is based on a low cost, high pressure ratio, un-cooled turbine which is designed to be scaled for turboshaft applications in the range of 50 to 300 kW. This paper describes the design of the base turbine, the scaling of this turbine which is required to produce the aerodynamic model and presents the results of an initial investigation using the model turbine.
THE DESIGN OF A HIGH-SPEED RADIAL INFLOW TURBINE

The baseline turbine duty was selected for a low cost single shaft gas turbine in the 200kW class. This results in modest gas temperature levels. However, the aerodynamic duty would also be appropriate for the gas generator turbine of a higher pressure ratio two shaft engine. The leading parameters are:

- Mass Flow: 1.58 kg/s
- Stator Inlet Temperature: 1161 K
- Total-Total Pressure ratio: 4.7
- Target Total-Total efficiency: 87%

**Preliminary Design**

The preliminary design process used many of the rules developed by NASA (e.g., Rohlik (1975) or Glassman (1976)). Several mechanical constraints were placed on the design to enable it to achieve an acceptable life with relatively low manufacturing costs, and to enable it to be scaled for other applications to approximately half size without significant modifications. The aerodynamic design method used was based on that of Glassman (1976). This method was linked to a preliminary geometry definition program and stress calculating routines.

![Preliminary Design Diagram](image)

**Figure 1: Meridional geometry and rotor inlet and exit rms velocity triangles for the full size turbine**

Rohlik (1975) gives the optimum incidence for this turbine as being -26°. For zero exit swirl, this would require a tip speed of about 650 m/s. This in turn defines the necessary blade taper ratio to achieve the required stress levels and hence the hub blockage. Figure 1 shows that -26° of swirl were left at the turbine rms exit radius in order to extract more work from the turbine exducer and hence minimise the tip speed. The incidence angle has also been compromised a little for the same reason resulting in a high rotor reaction, which should provide favourable flow conditions in the rotor. The rotor blade number was set to 14 with an axial length of 64 mm giving a maximum blockage at the hub of about 60%.

The stator blade number and chord were set using a lift coefficient defined as

\[ C_L = \frac{\pi r_1 V_{r1} - \pi r_2 V_{r2}}{Z \int (P_1 - P_2) r dr} \]

with a value of 0.6 being initially chosen. This is a modest value by axial turbine standards, and was chosen so that the secondary flows could be minimised. This was important as the aspect ratio of this vane is low compared to those of many axial turbines.

The Glassman model predicted a total-total efficiency of 84% for this preliminary turbine configuration, 3 points below the design target.

**Detail Design - Nozzle**

The detail design of the nozzle utilised standard Rolls Royce axial turbine blade design programs. A streamline curvature throughflow model of the turbine defined the stator and rotor inlet and exit conditions. The velocity distributions were calculated using a blade to blade program, and the final design was chosen by assessing the velocity distributions and the predicted boundary layer behaviour.

**Detail Design - Rotor**

At the outset it was realised that the flow in the rotor passages would be highly three dimensional and that it would be beneficial to design using three dimensional flow analysis programs.

Having estimated a meridional shape and blade thickness distribution, computed velocity distributions were obtained using the inviscid code of Denton (1983) and were assessed with the aim of minimising diffusion on the blade surfaces and avoiding flow reversal within the passages. The main region of difficulty appeared in the inducer region, where the high loading was leading to very high velocities on the suction surface at the shroud, and large levels of diffusion at the hub. This was not surprising, given that the number of blades employed was less than ideal.

Several alternative blade angle and annulus distributions were then assessed, along with the use of more blades, although the high levels of blockage necessitated by the blade taper made this an unattractive option. Finally the use of rake at blade inlet was explored, with the objective of applying blade lean to unload the shroud section. The final design employed 20° of rake at the inlet and figure 2 compares the final design Mach number distributions to an earlier attempt with no rake. It can be seen that the shroud velocity distribution has been markedly improved by applying the rake, although the hub diffusion has worsened. There is significant hub diffusion even in the earlier design, and it was therefore decided that little could be done to alleviate this and that as the majority of the work is done in the outer half of the rotor, it would be best to optimise the shroud distribution.

**AERODYNAMIC DESIGN OF LARGE-SCALE LOW-SPEED MODEL**

To obtain access to the flow within the passages of a radial inflow turbine and to permit high quality data acquisition, a machine larger than standard has to be built. It was decided at an early stage to construct a model turbine of a similar scale to that of the low-speed axial research facilities at the Whittle Laboratory, thus permitting the best possible resolution of the mainstream, boundary layer and secondary flows. This new rig would offer, for the first time, the opportunity to acquire data of a detail and quality similar to that so far obtained only in axial flow research programmes.

A large number of factors led to the selection of a rotor tip diameter of 1.219 m operating at 450 rpm. This gives approximately the correct Reynolds numbers for the stator and rotor but the peak Mach numbers are of the order of 0.1. It follows that true dynamic similarity cannot be maintained because the effects of compressibility cannot be simulated. The use of a low-speed rig therefore adds further complexity to the modelling process. Wisler (1984) and Joslyn et al (1990) both describe approaches to low-speed similarity modelling similar to those adopted here.

![Mach Number Distributions](image)

**Figure 2: Mach number distributions for the full size turbine showing the effects of rotor blade inlet rake**
Meridional Geometry

**Velocity triangles.** The work-mass flow characteristics of any turbine are effectively prescribed by the velocity triangles at inlet to and exit from the rotor blade. This means that it is essential that any model of the full-size turbine must have equivalent if not identical velocity triangles.

The rotor inlet radius \( r_3 \) is constant across the span so that the angular momentum entering the rotor is also constant. This is not generally true at rotor exit. If the flow conditions and work-mass flow characteristics of the model are to be matched to those of the full-size turbine at all flow coefficients along all streamlines, then the shroud inlet to shroud exit radius ratio \( (r_{shroud}/r_{hub}) \), the hub-shroud radius ratio at rotor exit \( (r_{shroud}/r_{hub}) \) and the radial variation of the swirl velocity \( V_{s}(r) \) must be scaled exactly.

Matching the exit flow field of the model rotor in all its details to the flow field behind the full-size rotor means that true matching can only occur at this location. Both the full-size and the model turbine operate at the same nominal specific speed, \( N_s \), of 0.6 which corresponds to the optimum value given by the NASA correlations and design rules (see Rohlik (1975)).

**Variation of streamtube height.** The variation of density is negligible at low-speed. Thus, in order to produce meridional velocity variations and, therefore, velocity triangles which are correctly scaled, the heights of the streamtubes would need to vary much less in the low-speed model than in the high-speed turbine. This meant that true geometric similarity could not be maintained. Given that the exit hub-to-shroud radius ratio \( (r_{shroud}/r_{hub}) \) was already prescribed, the desired variation in streamtube thickness could only be achieved by increasing the width (\( b_3 \)) of the inducer.

The width of the inducer in the model turbine was increased by a factor of about two relative to the scaled dimension. Although this is a large increase, the design is not significantly compromised. At rotor inlet, for example, the effects of secondary flow (see Zangeneh et al (1988)) and of tip clearance (see Futural and Holeski (1970)) are not significant as in the inducer at the design point, since large scale incidence induced separations will not occur.

**Rotor hub and shroud profiles.** The importance of streamline (meridional) curvature in the generation of secondary flows may be characterised by reference to the Rossby number

\[
R_0 = \frac{W}{\Omega c} \tag{2}
\]

where \( c \) is the local radius of curvature and \( W \) is the local relative velocity of the flow. Since the radius of curvature is significantly smaller and the loading significantly greater at the shroud than at the hub, it is not unreasonable to find that the secondary flows are more significant near the shroud. Hiett and Johnston (1963) and of tip clearance (see Futural and Holeski (1970)) are not as significant as in the inducer at the design point, since large scale incidence induced separations will not occur.

**Reynolds numbers.** Hiett and Johnston (1963) have reported the effect of changes in Reynolds number on the performance of radial inflow turbine rotors. The investigations showed that at Reynolds numbers, defined as

\[
Re = \frac{U s b_1}{v_1} \tag{3}
\]

above \( 1.25 \times 10^5 \), there is very little change in efficiency with Reynolds number.

Even though the flow accelerates over much of the blade span, consideration of the high magnitude of the surface length based Reynolds number shows that turbulent flow is already well established over much of the blade when operating above the limits established by Hiett and Johnston. The full-size turbine has a Reynolds number of \( 2.4 \times 10^5 \) which indicates that its operation is within the turbulent boundary layer regime.

The absence of a variation in density (and viscosity) through the model turbine and maintaining the nozzle radius ratio \( (r_1/r_2) \) would mean that the Reynolds number of the stator blades would be lower in the model by a factor of 0.6. In addition, the aspect ratio of the stator blades would also be greater by about the same factor due to the increased streamtube thickness at the rotor inlet. In an attempt to offset some of the effects of these potential changes, the chord and therefore the radius ratio \( (r_1/r_2) \) of the stator blades was increased while maintaining the same radius ratio \( (r_2/r_3) \) for the vanesless space and lift coefficient for the stator blades (equation 1). Increasing the radius ratio of the stator blades also reduces the impact of using parallel as opposed to opposing endwalls in the model since the velocity ratio \( V_2/V_1 \) is increased.

The Reynolds number of the stator blades, when defined as

\[
R_{estator} = \frac{P_2 V_2}{\nu} \tag{4}
\]

has a value of \( 2.8 \times 10^5 \) in the full-size turbine. When based on true chord, the value is \( 4.5 \times 10^5 \). This means that these stator blades are operating at conditions where the laminar-turbulent transition may well occur. Because the rotor of the full-size turbine operates well within the turbulent flow regime, the model turbine can be operated with the correct stator Reynolds number but with a higher rotor Reynolds number without significantly affecting the flow conditions in the rotor.

**Nozzle Guide Vanes.**

There are 23 NGVs in the model turbine which represents a reduction of six from the original 29 in the full-size. The majority of this decrease is simply due to the change in radial chord. The rotor inlet flow angle is the same in both cases. The effect of contouring the endwall in the full-size turbine can be seen by examining figure 3. This shows the meridional projection of the predicted suction surface Mach number contours obtained using the method of Denton (1983). The extent of the effects of the wall curvature is limited to less than one half of the span and that rather than being disadvantageous, the convex curvature of the curved endwall results in a greater amount of diffusion beginning near 75 percent radial. The parallel endwall has, by contrast, very little back-surface diffusion. These observations further support the original decision to employ parallel endwalls in the model turbine where the blade surface velocity distributions were matched to the Mach number distributions of the straight endwall of the full-size turbine. It may, of course, be argued that altering the endwall contours will lead to substantially different secondary flow fields in the model turbine. Figure 4 shows the predicted secondary flow patterns obtained using the flow prediction method of Dawes (1986). The figure shows that there is very little secondary flow in the model turbine so any changes that might occur will be very small.

**Integral boundary layer calculations were carried out using the method of Herbert and Calvert (1982) for the Mach number distributions of the straight endwall of the full-size turbine and for the model turbine.** These calculations showed that the momentum thickness based Reynolds number \( (Re_{\theta}=6V/\nu) \) was everywhere less than the critical value of 163 (see Herbert and Calvert (1982)), so that transition to turbulent flow would not occur. Because there was little or no diffusion, the boundary layers would also remain attached.

---

**Figure 3:** Meridional view of stator suction surface Mach number contours (Denton) showing the effect of wall curvature
Flow direction

--- Endwall Flow
--- Mid Passage Flow

Stator Endwall  Suction Surface

Figure 4: Predicted flow visualisation for the stator endwall and suction surface using the Dawes code

Rotor

Blade numbers. Stressing considerations dictate that the camberline of the rotor blades are formed from radial elements. Near to the leading edge, the rotor blades are radial. A simple analysis shows that for incompressible flow between radial blades of a many-bladed impeller, the velocity jump across the blades $\Delta W$ is given by

$$\frac{\Delta W}{W_m} = \frac{4\pi}{Z}$$

so that the loading is simply determined by the number of blades. Slip factors and therefore optimum inlet loading are also thought to be independent of Mach number below 0.8 (see Stanitz (1952)), further suggesting that the number of blades in the rotor of the model should be the same as in the full size machine.

The effects of blade lean at rotor inlet. In the high-speed turbine, the rotor leading edge is leaned so that the pressure side is towards the shroud. Because the density of the flow in the model turbine is effectively constant, the width of the inducer had been increased by a factor of about 2. This presented a difficulty as to the choice of the scaling parameters which should be used to specify the angle of lean in the low-speed rotor. A need for a simple analysis therefore existed. The analysis described in appendix A is applicable to the general case where shroud curvature exists at the leading edge plane but where there is no blade curvature. Figure A1 shows a plan and meridional view of the rotor blades in the region of the leading edge together with the leading dimensions of the problem. The result of the analysis shows that the important modeling relationship is:

$$\frac{b}{r} = \frac{2 \tan \theta}{W_m - r_c}$$

where all terms are defined in Appendix A.

The analysis is for the general case where there is meridional curvature. However, for the present study, the rotors under consideration have no meridional curvature at the leading edge and using the method described by Casey and Roth (1984) for analysing sweep the effective blade curvature is 1.6°. The blade lean has been applied solely to reduce the loading. The blade force is therefore unopposed by streamline curvature forces and the flow is deflected towards the hub. For similarity of flow conditions between the full-size and the model it follows that the value of $\theta / \tan \theta$ should be preserved. Applying the above equation to the adopted blade geometry for the model results in a 5% variation of absolute velocity across the span, corresponding to an incidence variation of 9°. Such a large variation in incidence onto a radial leading edge geometry is expected to increase losses but will be representative of existing radial inflow turbines and interesting to analyse.

Analysis of the rotor design using viscous flow predictions. For comparison between the predicted flowfields for the two machines, mid passage meridional and blade-blade relative Mach number contours are presented in figures 5 and 6. Although the two turbines are not identical the comparison appears to be reasonable.

Figure 5: Meridional mid passage relative Mach number contours

Figure 6: Mid passage blade-blade relative Mach number contours

Velocity vectors, in the meridional view, show the movements of the secondary flows for the different sections of the turbine (see figure 7). Comparison between the two turbines is good. As with the predictions of Zangeneh et al (1988), the secondary flow moves towards the hub on the pressure surface exducer and towards the shroud on the suction surface exducer under the influence of the reduced static pressure gradient. The effects of tip clearance flows may also be observed on both surfaces and in particular movement of low momentum flow up the exducer pressure surface towards the clearance region is apparent.

The comparison between the full size turbine and the model is thought to be adequate to provide a representative turbine on which to undertake aerodynamic research.

TEST FACILITY

The rig operates as an open loop wind tunnel drawing air from the atmosphere into the model turbine. The turbine exhausts into a circular duct which connects to a sliding throttle and the centrifugal fan. The open loop configuration with the suction fan after the working section ensures that axisymmetric, low turbulence flow can be obtained at entry to the blade row under investigation.

The stator blades were made from a glass reinforced epoxy resin cast from a machined mould. Each stator blade is mounted on two pins so that the stagger angle is correctly fixed. The measured variation of throat area corresponds to a variation of less than 0.2° in stator exit angle and 4° in rotor incidence. The rotor blades were manufactured by a slightly different process. A mould was again produced then a gel-coat of epoxy resin was placed in the mould and allowed to harden before a resin foam was injected into the mould. In this way light, rigid blades were obtained. Static pressure tappings were cast into both the stator and rotor blade surfaces.
is in agreement with the Herbert and Calvert (1982) predictions and satisfies the design objective of obtaining a stator with attached laminar boundary layers throughout.

The blade and endwall flow fields of the stator blade have been investigated using an oil and dye flow visualisation mixture. Figure 9 presents the resulting flow patterns. The upper figure shows the suction surface viewed looking upstream and may be compared to the predictions of figure 4. The lift off generated by entrainment of blade surface fluid to the passage vortex is apparent and shows the edge of the passage vortices to be at about 30% and 70% of the span. The lower photograph of the suction surface endwall corner region also shows a lift off line which will be due to the entrainment of endwall corner fluid within the passage vortex. There is no evidence of separation at any point on the suction surface. The flow visualisation is consistent with the predictions of figure 4.

Suction surface

Figure 8 shows the good comparison of experimental results from the stator blade statics with the predicted values from a Martensen method calculation at the design flow coefficient. The data shows that there is no diffusion on the suction surface which satisfies the design objective while the continuous acceleration on the pressure surface is a consequence of the relatively low loading coefficient chosen for this blade row.

Area traversing using a variety of probes can be carried out behind each blade row. The probes may be traversed over two or three pitches in the circumferential direction. A circumferential-radial traverse system is also fitted to the rotor at exit. This enables the investigation of the relative frame and in particular, of the internal passage flow. An on-rotor Scanivalve and pressure transfer system complements the rotor traverse system with slip-rings being used to carry information to the stationary frame.

DISCUSSION OF TEST RESULTS

Stator

Figure 8 shows the good comparison of experimental results from the stator blade statics with the predicted values from a Martensen method calculation at the design flow coefficient. The data shows that there is no diffusion on the suction surface which satisfies the design objective while the continuous acceleration on the pressure surface is a consequence of the relatively low loading coefficient chosen for this blade row.

![Predictions](image1)

Figure 8: Comparison between Martensen method prediction and experimental results for the radial stator blade

The state of the blade surface boundary layers has been determined by fitting surface-mounted hot film anemometers to both blade surfaces. Their operation and usage follows that described by Hodson and Addison (1988). The output signals from the gauges contained information characteristic of laminar boundary layers. This is in agreement with the Herbert and Calvert (1982) predictions and satisfies the design objective of obtaining a stator with attached laminar boundary layers throughout.

Rotor

The flow on the rotor blade surfaces has been investigated using a visualisation technique described by Joslyn and Dring (1983). At the design point, ammonia gas was passed out of the static pressure tappings in an instrumented blade, and allowed to flow over Ozalid paper. The results obtained using this method are shown in figure 11 and may be compared with the predictions of figure 7.

Area traversing has been undertaken in the absolute frame using fixed-direction 5-hole pyramidal pressure probes which have been calibrated for the measurement of yaw and pitch angles, total pressure and dynamic head. The results of traversing the stator exit flow field within the vaneless space are shown along with predicted total pressure contours in figure 10. In the measurements, the high loss regions associated with the passage vortices can be seen to be centred on approximately 20% and 80% span. Much of the flow remains unaffected by the secondary flow which is consistent with the flow visualisation patterns. The overall blade pressure loss coefficient (Yp) is 0.033 with a mid span value of 0.012. Compared to axial turbines the nozzle blade has a lower loss coefficient and appears to be performing well for an aspect ratio of 0.821 (based on radial chord). The reason for this level of performance is that both surfaces have attached laminar boundary layers over the whole length. For the prediction the vortices near the two suction surface endwalls observed during flow visualisation are apparent but are much smaller in extent than those measured. This is consistent with the differences between the predicted and measured surface flow patterns and is a consequence of the grid being too coarse to resolve the details of the flow accurately within the vortex.

![Rotor](image2)
most of the suction surface in the exducer, there are relatively strong secondary flows towards the shroud where the reduced static pressure is at a minimum. Very near to the shroud, the surface flow is directed toward the hub. The resulting herringbone pattern indicates the presence of a lift-off line. This feature is probably associated with the scraping vortex and suggests that the tip clearance vortex is relatively small in this machine.

The pressure surface flow visualisation of figure 11 shows that on the rotor, there is a large low momentum region close to the leading edge. Indeed, at the leading edge, some reversal of the flow direction may be observed. The reversed flow and pressure surface stagnation will be largely an inviscid effect and are to be anticipated since the turbine rotor is operating at an incidence which is greater than the Stanitz value. There is no evidence that this reversed flow region has a significant effect on the overall flow which is consistent with the observations of many researchers that reducing the blade numbers below the limit of pressure surface reversed flow does not necessarily inhibit the performance (e.g. Futral and Wasserbauer (1970)).

Downstream of this low momentum region, the flow responds to the meridional curvature and secondary movements occur towards the shroud. In the exducer, the influence of the reduced static pressure gradient means that the secondary flow is towards the hub as predicted by the Dawes code and by Choo and Civinskas (1985).

The results of traversing a 5-hole pyramidal probe at rotor exit are presented in figure 12. These plots show the mass flow-weighted pitchwise-average of the stagnation pressure loss and flow angle when the turbine is operating at the design flow coefficient. The overall total pressure loss coefficient for the turbine is 0.10. Using the mass averaged values of total pressure, the total-total efficiency derived using the Euler work equation is of the order of 93 percent. This value is approximately 5 percent higher than expected, a result which may be attributed to the difficulty of measuring the stator and rotor exit flow angle and mass flow distributions with sufficient accuracy. These problems will be resolved for future work.

The spanwise variations in total pressure loss ($Y_p$) and yaw angle shown in figure 12 follow the trends observed in other radial turbines (e.g. Futral and Holeski (1970)), except at the shroud where the loss is reduced and the yaw angle indicates the overturning which is consistent with the existence of a scraping vortex. These differences are probably due to the relatively small tip clearance of the model turbine. If this is indeed the case, then the region of high loss near to the shroud is probably due to the accumulation of suction surface low momentum fluid and is not a direct result of shroud clearance flows. This observation, which is consistent with the rotor flow visualisation (figure 11), is in agreement with the suggestions of Zangeneh et al (1988). Future investigations will reveal the true origins of this region of loss.

CONCLUSIONS

A gas generator radial inflow turbine has been designed to operate at a total-total pressure ratio of 4.7. This turbine has then been modeled and placed in a large scale, low speed test rig at the Whittle Laboratory.

The model stator blades show only weak secondary flow movements and have laminar blade surface boundary layers over the whole blade surface. The overall loss coefficient based on rotor inlet dynamic head is 0.033.

The rotor blades show strong secondary flow movements on both surfaces and a low momentum region on the pressure surface at inlet when at the design flow coefficient. Exit traverses display similar results to those obtained by other organisations but lower loss and yaw angle is to be observed towards the casing of the machine, suggesting that high shroud losses are due to an accumulation of low momentum fluid and not due to strong tip clearance flows.

A simple model has been developed to determine the effects of rotor blade inlet lean.

ACKNOWLEDGEMENTS

The authors would like to acknowledge Rolls Royce plc for support of the research work and also Rolls Royce Business Ventures Ltd for initiating and funding the original high speed turbine design work.
REFERENCES
Choo, Y K and Civinskas, K C, 1985, "Three dimensional inviscid analysis of radial turbine flow and a limited comparison with experimental data", NASA TM-8709
Dawes, W N, 1986, "A numerical analysis of the three dimensional viscous flow in a transonic compressor rotor and comparison with experiment", ASME paper 86-GT-16
Futral, S M and Holeski, D E, 1970, "Experimental results of varying the blade shroud clearance in a 6.02 inch radial inflow turbine", NASA TN D-5513
Glassman, A J, 1976, "Computer program for design analysis of radial inflow turbines", NASA D-8164
Herbert, M V Calvert, W J, 1982, "Description of an integral method for boundary layer calculation in use at NGTE, with special reference to compressor blades", NGTE memorandum MR82019
Rohlik, H E, 1975, "Radial Inflow Turbines", NASA SP 290, Vol 3, Chapter 10
Stanitz, J D, 1952, "Some theoretical aerodynamic investigations of impellers in radial and mixed flow centrifugal compressors", *Trans ASME*, vol 74, no 4, p473-497
Wisler, D C, 1984, "Loss reduction in axial flow compressors through low speed model testing", ASME paper no. 84-GT-184

APPENDIX A: A SIMPLE ANALYSIS OF THE EFFECTS OF INLET BLADE LEAN
Figure A1 shows a plan and meridional view of the rotor blades in the region of the leading edge together with the leading dimensions of the problem. The pressure gradient normal to curved streamlines is given by the general expression
\[
\frac{dp}{dn} = \frac{\rho W_{m}^{2}}{rc} \tag{A1}
\]
where \(rc\) is the radius of curvature and \(W_{m}\) is the meridional (i.e. mean) velocity. For large \(rc\) this expression can be approximated to
\[
\Delta p = \frac{\rho W_{m}^{2}}{rc} b \tag{A2}
\]
where \(\Delta p\) is the pressure difference between the hub and the shroud due to streamline curvature. The axial force developed by this pressure difference is:
\[
F_{c} = 2\pi r\Delta \Delta p = 2\pi r\Delta \frac{\rho W_{m}^{2}}{c} b \tag{A3}
\]

Leaning the blade results in a component of the blade force which acts towards the shroud. This component of force which is exerted by the fluid on the blades is given by
\[
F_{b} = b \tan \theta \Delta \Delta p Z \tag{A4}
\]
where \(Z\) is the number of blades and the blade pressure difference is given by the approximate expression
\[
\Delta \Delta p = \rho W_{m} \Delta W_{b} = \frac{4\pi}{Z} b U_{W_{m}} \tag{A5}
\]
from equation 5.

It is interesting to note here that if the two forces are equal, then the two effects cancel and so in the inducer, the flow is uniform regardless of the presence of the stream-line curvature and blade lean. This probably explains why turbochargers, which often have both lean and shroud curvature perform better than might be expected even though the shroud radius ratio is greater than the value of 0.7 recommended by the NASA design rules.

When the two forces do not balance, there will be a spanwise variation in the flow properties. Where there is very little curvature, there may be significant variations in the flow properties. The difference between the forces \(F_{c}\) and \(F_{b}\) will result in a reaction force \(F_{x}\), which will be related to the overall hub-to-shroud pressure difference \(\Delta p\) and velocity difference \(V_{hub} - V_{shroud}\) by the expression
\[
F_{x} = 2\pi r\Delta \Delta p = 2\pi r\Delta \rho \frac{\left( V_{hub} - V_{shroud} \right)}{W_{m}^{2}} = F_{b} - F_{c} \tag{A6}
\]
so that upon substitution from equations (A3), (A4) and (A5)
\[
\frac{V_{hub} - V_{shroud}}{W_{m}^{2}} = b\left( \frac{2 \tan \theta}{r_{c}} \frac{U_{W_{m}}}{r_{c}} \right) \tag{A7}
\]
from which the effects of blade lean and shroud curvature can be estimated.

![Figure A1: Determination of the effects of blade inlet rake](attachment:image.png)